

Running OpenFOAM on Android – A Workflow Tutorial With a Simple Case Study

Komal Kedarnath G

Mechanical Engineer, Researcher

Kolar Gold Fields, Karnataka, India

ABSTRACT:

Linux is a very powerful platform. It is quite a need these days to get accustomed to the usage of Linux with the advent of many open-source tools gaining traction. One such tool is OpenFOAM. Learning the basics of OpenFOAM can be achieved on a simple computer but with the increasing complexity of the problems, there is a demand for much more computational power that one can satisfy either through powerful computers or distributed cloud services (HPCs). This becomes a challenge in the early learning process for those with fewer privileges.

Android is at its core “just another Linux distribution”. Anything you can do on a typical Linux desktop can be achieved on android too. The technology is evolving as well to diminish the gap between handheld smartphones and desktop personal computers. It is therefore common to find a large population in India owning Android phones with at least 4 cores but don’t own computers that are equally or more powerful. This tutorial is intended to set a framework that can be adopted by many and built upon. A good chunk of the student population in India can benefit from this. Students who wish to learn CFD but can’t afford powerful computers fail to realize that they have a very powerful device in their hands. They can run effective cases locally on the device. For more powerful cases they can utilize SSH and terminal to access HPCs or Cloud services to run more advanced computations.

This tutorial will consist of two parts – 1. Setting up Android devices with as few extra applications as possible to run OpenFOAM. 2. Running a simple flow in elbow case in it. A few extended applications will be hinted upon for moderately advanced users. Anyone with knowledge of basic Linux can pick up on them to further their exploration and learn through it.